REPORT DOCUMENTATION PAGE

AFRL-SR-AR-TR-04-

0092

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for maintaining the data needed, and completing and reviewing this collection of information. Send comments regarding this b~~urden~~ suggestions for reducing this burden to Washington Headquarters Services, Directorate for Information Operations and Rep ~~-4302,~~ and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE 14 Jan 2004	3. REPORT TYPE AND DATES COVERED Final Report, 1 Sep 1999 – 1 Sep 2002	
4. TITLE AND SUBTITLE APPLICATION OF NEW ALGORITHMS FOR THE SIMULATION OF MULTIDISCIPLINARY PROBLEMS: FLUID, STRUCTURE, THERMAL AND CONTROL COUPLING		**5. FUNDING NUMBERS** G F49620-00-1-0015	
6. AUTHOR(S) Rainald Lohner and Chi Yang			
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) George Mason University MS 4C7, SCS 4400 University Drive Fairfax, VA 22033		**8. PERFORMING ORGANIZATION REPORT NUMBER** 5-25024	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) Air Force Office of Scientific Research Bolling Air Force Base Dr. Leonidas Sakell Washington, D.C. 20332-6448		**10. SPONSORING / MONITORING AGENCY REPORT NUMBER**	

11. SUPPLEMENTARY NOTES

20040213 113

12a. DISTRIBUTION / AVAILABILITY STATEMENT No limitation	12b. DISTRIBUTION CODE

DISTRIBUTION STATEMENT A
Approved for Public Release
Distribution Unlimited

13. ABSTRACT *(Maximum 200 Words)*

The overall objective of the research carried out over the last three years was the development of new algorithms for the efficient simulation of multidisciplinary problems that require the simulation of viscous, conducting, compressible flows around or within moving, structurally and thermally responding structures. The development was based on current 3-D Euler/Navier-Stokes capabilities, and encompassed flow solvers, grid generation, CFD-CSD-CTD integration, optimal shape design, and the efficient use of emerging super computer hardware. The research carried out over the last three years significantly advanced the state of the art in this area of CFD.

14. SUBJECT TERMS CFD, CSD, CTD, Fluid-Structure Interaction, Loose Coupling			15. NUMBER OF PAGES 20
			16. PRICE CODE
17. SECURITY CLASSIFICATION OF REPORT	**18. SECURITY CLASSIFICATION OF THIS PAGE**	**19. SECURITY CLASSIFICATION OF ABSTRACT**	**20. LIMITATION OF ABSTRACT**

FINAL REPORT: DECEMBER 2002

APPLICATION OF NEW ALGORITHMS FOR
THE SIMULATION OF MULTIDISCIPLINARY PROBLEMS:
FLUID, STRUCTURE, THERMAL AND CONTROL COUPLING

Rainald Löhner and Chi Yang
School of Computational Sciences
George Mason University
Fairfax, VA 22030-4444

presented to:

Air Force Office of Scientific Research /NA
c/o Dr. Leonidas Sakell
Bolling Air Force Base
Washington, D.C. 20332-6448

APPLICATION OF NEW ALGORITHMS FOR THE SIMULATION OF MULTIDISCIPLINARY PROBLEMS: FLUID, STRUCTURE, THERMAL AND CONTROL COUPLING

Rainald Löhner and Chi Yang
School of Computational Sciences
George Mason University
Fairfax, VA 22030-4444

Summary

The overall objective of the research carried out over the last three years was the development of new algorithms for the efficient simulation of multidisciplinary problems that require the simulation of viscous, conducting, compressible flows around or within moving, structurally and thermally responding structures. The development was based on current 3-D Euler/Navier-Stokes capabilities, and encompassed flow solvers, grid generation, CFD-CSD-CTD integration, optimal shape design, and the efficient use of emerging super computer hardware. The research carried out over the last three years significantly advanced the state of the art in this area of CFD. The particular topics are treated below in detail.

1. FLOW SOLVERS

For the flow solvers, seven major developments took place over the course of this research effort:

- Incompressible solver modules:
    - New implicit-advection projection solver;
    - Pressure boundary conditions for internal flows;
    - Arbitrary number of advective/diffusive species;
    - Inclusion of vorticity confinement to track vortices over large distances;
    - Combination of Baldwin-Lomax and Smagorinsky turbulence models;
- Compressible solver modules:
    - Two matrix-free implicit compressible solvers;
    - Arbitrary-gas equation of state lookup;
    - Combustion modeling.

1.1 Incompressible Flow Solvers

New Implicit-Advection Projection Solver: To date, the advective terms present in the Navier-Stokes equations have been integrated explicitly. This yields highly accurate results, but also imposes severe restrictions on the allowable timestep. For grids that exhibit a large variation in element size the timestep imposed by the smallest elements may be orders of magnitude smaller than the timestep required to obtain time-accurate results. This implies that for many classes of problems, e.g. heat release with buoyancy effects, tens of thousands of timesteps are required per simulation, rendering the scheme impractical. This prompted the quest for an implicit integration of the advective terms that could be incorporated easily into the existing, mature, explicit-advection projection scheme.

A class of implicit projection schemes was developed that allows to circumvent the stringent explicit advection timestep limitation without incurring a major code change. The core solver (explicit advection with local timestepping) is maintained intact, with the exception of simple source-terms that are pointwise and therefore themselves well for implicit integration. The implicit projection schemes were implemented into FEFLO, a general purpose CFD code based on unstructured-grids and simple finite elements. The implicit projection schemes were then used to solve transient flows, some of which had thermal buoyancy effects. The results show considerable savings for the implicit schemes as compared to their explicit advection counterparts.

For an application which is a time-accurate run, the new implicit scheme runs approximately 10 times faster than the explicit scheme. The option of advecting an arbitrary number of species was added to simulate the effects of contaminants, combustion and for diagnostics purposes.

The new scheme has been tested for a number of benchmark cases. Some of the test cases are listed as follows:

- VonKarman vortex street, cooling tower;

3

- Natural convection of air in a square cavity
- Cooling tower;
- Steady ship wave around Wigley hull.

Pressure Boundary Conditions For Internal flows: For external aerodynamics, the imposition of pressure at outflow boundaries presents no further difficulties. The situation is very different for internal flows, such as pipe networks or arteries in the human body. Two new pressure boundary conditions for internal flows were derived and implemented in FEFLO.

It was shown in the Circle of Willis in the brain the new pressure boundary conditions were the only way to simulate with some degree of realism the complex hemodynamic patterns that arise.

Vorticity Confinement: A new, general vorticity confinement term has been derived using dimensional analysis. The resulting vorticity confinement term is a function of the local vorticity-based Reynolds number, the local element size, the vorticity and the gradient of the vorticity. The vorticity confinement term disappears for vanishing mesh size, and is applicable to unstructured grids with large element size disparity.

The new vorticity confinement term was implemented into FEFLO. Different vorticity confinement options have been tested and compared, and the best was selected. The new term has been found to be successful for a number of test cases, allowing better definition of vortices without any deleterious effects on the flow field.

The verification and validation are carried out for the following cases:
- von-Karman vortex street;
- Wing tip vortex separation;
- Wing vortex interaction; and
- Massively separated flow over a wing.

The vortex core visualization for the NACA0012 wing showed that the vortex was well captured till eight chord lengths downstream of the trailing edge when the vorticity confinement term was switched on. The vortex was dissipated after only one chord length without vorticity confinement. The vorticity confinement also had a marked effect on the strength of the detached vortex for the delta wing case, and the strength of the von-Karman vortex for the 2-D cylinder. The large scale demonstration was performed for a submarine with a 42-foot diameter. The speed was 7 knots and the Reynolds number was $1.1x10^6$.

Baldwin-Lomax-Smagorinsky Model: The path to high-fidelity, high Reynolds-number simulations always leads to impossibly high numbers of gridpoints if all temporal and spatial wavelengths present in the flow are resolved. In order to obtain tractable numbers of gridpoints, either temporal averaging (via Reynolds-Averaged Navier-Stokes (RANS)) or some form of spatial averaging (via Large Eddy Simulations (LES)) is required. A relative new and promising approach is Detached Eddy Simluation (DES), which merges the best of RANS and LES. RANS is used near the wall where there is no separation. LES is used in the core flow where it works well. In this spirit, the two

simplest algebraic models: Baldwin Lomax and Smagorinsky were combined in such a way that each one is used in the region of the flow it was devised for. The Baldwin Lomax (BL) turbulence model is used close to walls, where it is known to give very good results. Smagorinsky (SMA) is used for regions of separation and core flow. In a first pass, the SMA viscosity is computed from the vorticity or second invariant of the deformation rate tensor, as well as the local mesh size. In a second pass, the BL viscosity is computed as usual, i.e. by dividing the boundary layer into an inner and outer layer. For the inner layer, the BL viscosity is kept. In the outer layer, the BL viscosity is used until it falls below the value of the SMA viscosity. In the remainder of the flow domain, the SMA viscosity is used.

1.2 Compressible Flow Solvers

Two Matrix-Free Implicit Compressible Solvers: If we consider a steady flow problem, the fastest solvers to date include multigrid and GMRES-LU-SGS solvers. For supersonic flows, space-marching and blocking represent an interesting alternative. During the past three years, the GMRES-LU-SGS solvers developed in prototype versions of FEFLO were migrated into the production code.

The classic relaxation schemes to solve the resulting system have been optimized over the years, resulting in very efficient solvers. The combined effect of these simplifications is a family of schemes that are matrix free, require no extra storage as compared to explicit schemes, and per relaxation sweep are faster than conventional explicit schemes.

Abitrary-Gas Equation of State Lookup: In order to be able to model effectively mixtures of gases such as those encountered in nozzle exhaust plumes, a conversion code was written for the NASA-Glenn chemical equilibrium program CEA of Gordon and McBride. The resulting table is output in the Los Alamos SESAME format, and read in at the beggining of the run. The table look-up requires less than 5% of the total CPU time for explicit solvers on RISC machines, but has not yet been ported to vector machines.

Combustion Modeling: During past three years, we implemented both equilibrium and finite rate models. The core modules follow closely the techniques of CONCHAS and KIVA. The stiff ODE integrator decomposes the reactions into fast and slow ones, and applies different integration techniques depending on the speed.

2. GRID GENERATION

In the area of grid generation, there were five major developments that took place during the course of this research effort:
- Advances in the gridding of surfaces given as discrete data;
- Improvements in RANS gridding;
- Improvements in parallel grid generation.

2.1 Discrete Surface Meshing

The rapid, user-friendly definition of the surfaces defining the computational domain has been an important goal during the last decade. Surfaces can be defined either analytically (using B-Splines, NURBS, Coon's patches, etc.) or via triangulations. The latter option is particularly interesting for data sets stemming from remote sensing data (e.g. geographical data) or medical imaging. An interesting observation made over the last years is that an increasing number of data sets used to define the geometry of CFD domains is given in the form of triangulations, even though the CAD data is available. The reason for this shift in data type is that a watertight triangulation defines in a unique way the domain considered, and does not require any futher geometric cleanup operations. This is not the case with native CAD datasets, in which we frequently encounter very large numbers of patches, overlapping patches, gaps, and other geometric pathologies that require user intervention. These developments have renewed the interest in robust surface meshing of so-called discrete surfaces (DS). Of the many innovations introduced during the last years, we mention:
- Automatic preprocessing/improvement of the DS;
- Introduction of a visibility horizon filter for close points/ sides;
- Strict enforcement of continuous topology;
- Improved 2D cross-check; and
- Adaptive background grid element size definition.

In the sequel, we expand on a few of these.

Visibility Horizon: The advancing front method adds a new surface triangle by removing a side from the active front. Among the decisions required is whether to take an existing point to form the new triangle, or to introduce a new point. The list of close (i.e. possible) points is obtained from a proximity search. This list of possible close points is reduced by several tests (visibility, angles, etc.). Perhaps the most important validation test is based on the neighbour to neighbour search on the given DS. The starting face for the search is given by the underlying DS face at the midpoint of the side being removed from the active front. The direction is given by the close point. Any close point that can not be reached on the given DS using the neighbour to neighbour search is removed from the list. A similar procedure is used to filter close sides, which are required to test if the new triangle crosses the existing active front of sides.

Continuous Topology: A typical neighbour to neighbour search will not stop at internal DS geometry lines (given by sharp edges). Therefore, a so-called visibility horizon was introduced for the neighbour to neighbour search. All neighbour to neighbour edges given by internal geometry lines or angles beyond a certain tolerance are marked. In this way, the neighbour to neighbour search can recognize them. The neighbour to neighbour search stops at these internal geometry lines. The close point is marked as unreachable and removed from the list of candidates.

Adaptive Background Grid Based on DS: For complex geometries, the specification of desired element size can be a tedious, time-consuming process. Adaptive background grids, offer the possibility to reduce drastically the required level of human input. DS

6

offer, by their way of defining the surface, a natural way to refine the background grid and to define the mesh size required for a proper definition of the geometry.

2.2 Improvements in RANS Gridding

The generation of high-quality grids suitable for RANS calculations of flows in and around complex geometries continues to be an active area of research. The two most common ways of generating highly stretched grids suitable for RANS calculations are the advancing layers technique (ALT) and the directional enrichment technique (DET). The ALT follows the spirit of the advancing front technique (AFT): starting from the 'wetted surface', add thin layers of elements until an isotropic mesh is achieved. From this point onwards, the mesh is completed with the AFT. The DET is an extension of the Generalized Delaunay Triangulation (GDT). At first, an isotropic mesh is generated (either via AFT or GDT). Then, the points in the regions where stretched elements are to be generated are removed (typically by collapsing edges). Thereafter, points are introduced in the near-wall region so as to obtain the desired RANS mesh. The newly introduced points are reconnected using the GDT. This latter technique, while general, can still generate bad grids for complex geometries. A recent improvement was obtained by considering the layer-number of the points when performing diagonal/face swapping in 3-D. The quality of elements that cross several point-layers is reduced, leading to swaps that favour elements with minimal layer jump.

2.3 Parallel Grid Generation

Over the last decade, major efforts have been devoted to harness the power of parallel computer platforms. While many CFD and CSD solvers have been ported to parallel machines, grid generators have lagged behind. For applications where remeshing is an integral part of simulations, e.g. problems with moving bodies or changing topologies, the time required for mesh regeneration can easily consume more than 50% of the total time required to solve the problem. Faced with this situation, a number of efforts have been reported on parallel grid generation.

The two most common ways of generating unstructured grids are the Advancing Front Technique (AFT), and the Generalized Delaunay Triangulation (GDT). The AFT introduces one element at a time, while the GDT introduces a new point at a time. Thus, both of these techniques are, in principle, scalar by nature, with a large variation in the number of operations required to introduce a new element or point. While coding and data structures may influence the scalar speed of the 'core' AFT or GDT, one often finds that for large-scale applications, the evaluation of the desired element size and shape in space, given by background grids, sources or other means consumes the largest fraction of the total grid generation time. Unstructured grid generators based on the AFT may be parallelized by invoking distance arguments, i.e. the introduction of a new element only affects (and is affected by) the immediate vicinity. This allows for the introduction of elements in parallel, provided that sufficient distance lies between them.

7

A convenient way of delimiting the possible zones where elements may be introduced by each processor is via boxes. These boxes may be obtained in a variety of ways, i.e. via bins, binary recursive trees, or octrees. We have found the octree to be the best of these possibilities, particularly for grids with a large variation of element size. In order to recover a parallel gridding procedure that resembles closely the advancing front technique on scalar machines, only the boxes covering the active front in regions where the smallest new elements are being introduced are considered.

At the end of each parallel gridding pass, each one of the boxes gridded can have an internal boundary of faces. For a large number of boxes, this could result in a very large number of faces for the active front. This problem can be avoided by shifting the boxes slightly, and then regridding them again in parallel. This simple technique has the effect of eliminating almost all of the faces between boxes with a minor modification of the basic parallel gridding algorithm.

3. CFD-CSD-CTD Integration

Fluid-Structure-Thermal Interaction simulations represent one of many possible interdisciplinary application areas. One way to solve such interdisciplinary problems in a cost-effective manner is via the so-called loose coupling algorithm. Each core discipline code is changed as little as possible, and the transfer of information between regions or surfaces is carried out by an independent general-purpose library. Results from a vast range of simulations carried out over the last decade indicate that the proposed approach offers a convenient and cost-effective way of coupling computational fluid dynamics (CFD), computational structural dynamics (CSD), computational thermal dynamics (CTD) codes without a complete re-write of them.

The most important question is how to combine these different disciplines to arrive at an accurate, cost-effective, and modular simulation approach that can handle an arbitrary number of disciplines at the same time. It is clear that any multidisciplinary capability must have the ability to quickly switch between approximation levels, models, and ultimately codes, and only those approaches that allow a maximum of flexibility, i.e.:
- Linear and nonlinear CFD, CSD and CTD models;
- Different, optimally suited discretizations for CFD, CSD and CTD domains;
- Modularity in CFD, CSD and CTD models and codes;
- Fast multidisciplinary problem definition; and
- Fully automatic grid generation for arbitrary geometrical complexity.

We developed a loose coupling approach, which includs the direct coupling in time of explicit CFD and CSD codes and the incremental load approach of steady aero- and hydro-elasticity. The variables on the boundaries are transferred back and forth between the different codes by a master code that directs the multi-disciplinary run. Each code (CFD, CSD, CTD, ..) is seen as a subroutine, or object, that is called by the master code, or as a series of processes that communicate via message passing. This implies that the transfer of geometrical and physical information is performed between

8

the different codes without affecting their efficiency, layout, basic functionality, and coding styles. At the same time, different CSD, CTD or CFD codes may be replaced, making this a very modular approach. This allows for a straightforward re-use of existing codes and the choice of the 'best model' for a given application. The information transfer software may be developed, to a large extent, independently from the CSD, CTD and CFD codes involved, again leading to modularity and software reuse. For this reason, this approach is favoured for industrialization. Indeed, cosiderable effort has been devoted to develop general, scalable information transfer libraries.

For the CFD-CSD-CTD integration, we have made great improvements on the coupling schemes and information trasfer, including surface to surface interpolation and load and flux transfer, over the course of this research effort.

4. TREATMENT OF MOVING SURFACES/BODIES

Any fluid-structure interaction simulation with considerable deformation of the structure will require a flow solver that can handle the arbitrary deformation of surfaces in time. The treatment of these moving surfaces is different depending on the mesh type chosen. For **body-conforming** grids the external mesh faces match up with the surface (body surfaces, external surfaces, etc.) of the domain. This is not the case for the **embedded approach** (also known as ficticious domain, immersed boundary or Cartesian method), where the surface is placed inside a large mesh (typically a regular parallelepiped), with special treatment of the elements close to the surfaces. Considering the general case of moving or deforming surfaces with topology change, both approaches have complementary strengths and weaknesses:

a) Body-Conforming Moving Meshes: the PDEs describing the flow need to be cast in an arbitrary Lagrangean-Eulerian (ALE) frame of reference, the mesh is moved in such a way as to minimize distortion, if required the topology is reconstructed, the mesh is regenerated and the solution reinterpolated. All of these steps have been optimized over the course of the last decade, and this approach has been used extensively. The body-conforming solution strategy exhibits the following shortcomings: the topology reconstruction can sometimes fail for singular surface points; there is no way to remove subgrid features from surfaces, leading to small elements due to geometry; reliable parallel performance beyond 16 processors has proven elusive for most general-purpose grid generators; the interpolation required between grids invariably leads to some loss of information; and there is an extra cost associated with the recalculation of geometry, wall-distances and mesh velocites as the mesh deforms. On the other hand, the imposition of boundary conditions is natural, the precision of the solution is high at the boundary, and this approach still represents the only viable solution for problems with boundary layers.

b) Embedded Fixed Meshes: the mesh is not body-conforming and does not move. Hence, the PDEs describing the flow can be left in the simpler Eulerian frame of reference. At every timestep, the edges crossed by CSD faces are identified and proper boundary conditions are applied in their vicinity. While used extensively this solution

9

strategy also exhibits some shortcomings: the boundary, which has the most profound influence on the ensuing physics, is also the place where the worst elements are found; at the same time, near the boundary, the embedding boundary conditions need to be applied, reducing the local order of approximation for the PDE; no stretched elements can be introduced to resolve boundary layers; adaptivity is essential for most cases; and there is an extra cost associated with the recalculation of geometry (when adapting) and the crossed edge information.

Extensions and improvements of an adaptive embedded unstructured grid technique are made over past three years. These include: improvements in **speed** via suitable data structures, treatment of **multimaterial problems**, in particular water/air, the option to treat dispersed **particles** in the context of embedded surfaces, a direct link to **Discrete Particle Methods (DPM)**, a volume to surface meshing technique that obtains body-fitted grids by post-processing adaptive embedded grids; and links to simplified **CSD models**. Numerous examples demonstrate the techniques developed.

Prompted by the need to solve shock-structure interaction problems and the inability of Computational Structural Dynamics (CSD) codes to ensure strict no-penetration during contact, a simple embedded technique operating on adaptive, unstructured grids was implemented. The essential elements of this technique may be summarized as follows:

a) The key modification of the original, body fitted edge-based solver was the removal of all geometry-parameters (essentially the area normals) belonging to edges cut by embedded surface faces.

b) Several techniques to improve the treatment of boundary points close to the immersed surfaces were implemented. Higher-order boundary conditions were also achieved by duplicating crossed edges and their endpoints.

c) Geometric resolution and solution accuracy were enhanced by adaptive mesh refinement that was based on the proximity to or the curvature of the embedded CSD surfaces.

d) In order to save work, user-defined or automatic deactivation for regions inside immersed solid bodies was employed.

The technique has been extremely successful for complex fluid-structure interaction cases, as well as some external aerodynamics cases.

5. OPTIMAL SHAPE DESIGN

The relentless advance in numerical methods and computer power has made accurate flow simulations of realistic geometries a reality. Such simulations are increasingly reducing the amount of lengthy (and costly) experiments in the aerospace, car, train and shipbuilding industries, substituting them for high fidelity CFD runs. This way of utilizing CFD is nothing more than an exchange of real for virtual experiment. However, CFD and its underlying mathematics offers the possibility to step beyond the capabilities of any experiment. While the experiment (or stand-alone CFD run) only measures the performance of the product 'as is', numerical methods can also

predict the effect of changes in the shape of the product. This has led, over the last decade, to a large body of literature on optimal shape design.

A complete CFD design methodology is investigated. The main components of this methodology are: a) a general edge-based compressible/incompressible flow solver, b) a continuous adjoint formulation for the gradient computations, c) a steepest descent technique for the change of design variables, d) evaluation of the gradient of the discretized flow equations with respect to mesh by finite differences; e) a CAD-free pseudo-shell surface parametrization, allowing every point on the surface to be optimized to be used as a design parameter, and f) a level type scheme for the movement of the interior points.

An optimal shape design procedure based on the solution of the continuous adjoint has been implemented in FEFLO. Several results, spanning compressible and incompressible, viscous and inviscid flow, demonstrate the usefulness of the capabilities developed.

6. EFFICIENT USE OF SUPERCOMPUTING HARDWARE

Despite the striking successes reported to date, only the simplest of all solvers: explicit timestepping or implicit iterative schemes, perhaps with multigrid added on, have been ported without major changes and/or problems to massively parallel machines with distributed memory. Many code options that are essential for realistic simulations are not easy to parallelize on this type of machine. Among these, we mention local and global remeshing, repeated h-refinement, such as required for transient problems, contact detection and force evaluation, some preconditioners, applications where particles, flow, and chemistry interact, fluid-structure interaction with topology change and, in general, applications with rapidly varying load imbalances. Even if 99% of all operations required by these codes can be parallelized, the maximum achievable gain will be restricted to 1:100. If we accept as a fact that for most large-scale codes we may not be able to parallelize more than 99% of all operations, the shared memory paradigm, discarded for a while as non-scalable, will make a comeback. It is far easier to parallelize some of the more complex algorithms, as well as cases with large load imbalance, on a shared memory machine. And it is within present technological reach to achieve a 100 processor, shared memory machine (128 has been a reality since 2000).

7. EXAMPLES

The loose coupling methodology has been applied over the last five years to a number of problems. We include here some recent examples, going from simple rigid-body CSD motion to highly nonlinear, fragmenting (i.e. topology-changing) solids. For more examples, including validation and comparison to experiments.

7.1 Series-60 Hull: The first example considers the steady (incompressible) flow past a typical shiphull. The hull is allowed to sink and trim due to the fluid forces. The final position and inclination (trim) of the hull are obtained iteratively. In each iteration, the steady flow is computed, the forces and moments evaluated, and the ship repositioned.

11

The mesh is typically moved. Should the need arise, a local or global remeshing is invoked to removed elements with negative volumes. The geometry considered is shown in Figure 1a. The mesh consisted of approximately 400,000 elements. Figures 1b,c depict the convergence of the computed sinkage and trimm with respect to the number of iterations. Figures 1d,e present a comparison of the computed sinkage and trimm with experimental data. Figures 1f,g show a comparison of the computed wave drag coefficient with experimental data for the model fixed and free to sink and trimm respectively. A run of this kind can be obtained in less than an hour on a leading-edge PC.
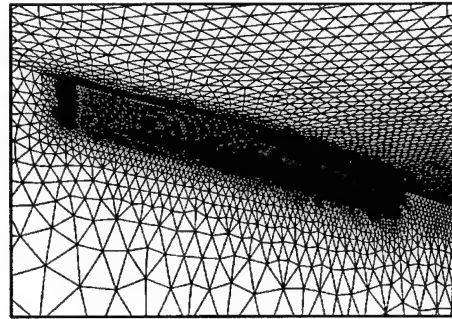
Figure 1a Series 60 Hull: Surface Mesh
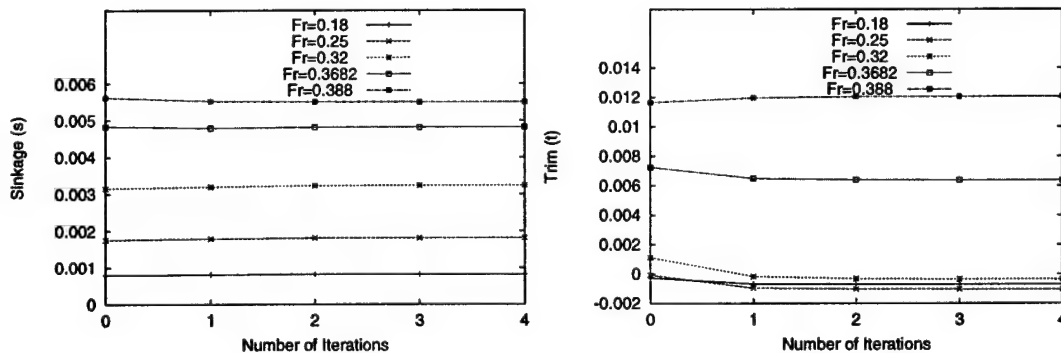
Figure 1b,c Series 60 Hull: Convergence of Sinkage and Trim
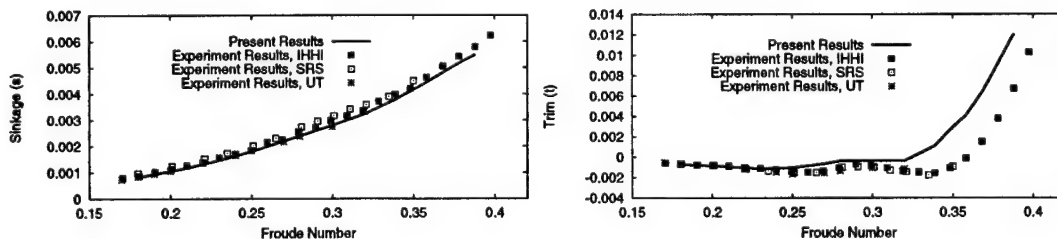
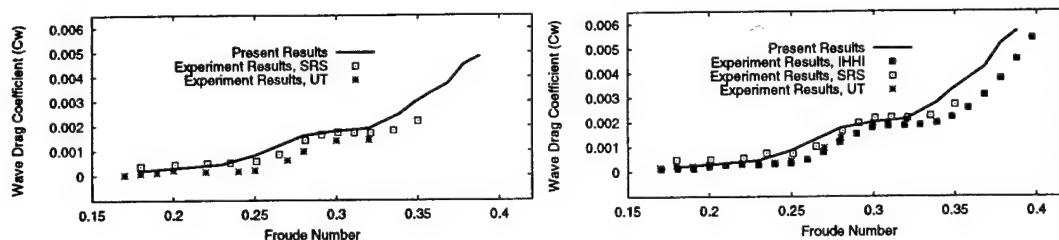Figure 1d,e Series 60 Hull: Sinkage and Trim vs. Froude-Nr.

12

Figure 1f,g Series 60 Hull: Wavedrag for Fixed and Free Model

7.2 <u>Nose-Cone</u>: Figure 2 shows results for a proposed nose-cone experiment. The CFD part of the problem was computed using FEFLO98, and the CSD and CTD with COSMIC-NASTRAN. Details of the flow solver may be inferred from. The incoming flow was set to $M_\infty = 3.0$ at an angle of attack of $\alpha = 10^o$. The Reynolds number was approximately $Re = 2 \cdot 10^6$, based on the length of the cone. The solution was initiated by converging the fluid-thermal problem, without any structural deformation. Thereafter, the fluid-structure-thermal problem was solved. Converegence was achieved after 10 cycles. It is interesting to note that the converegence is markedly slower than that achieved for fluid-structure (i.e. aeroelastic) problems. This is due to the interplay of temperature advection in the flow domain and conduction in the solid, whose counteracting effects have to be balanced out.
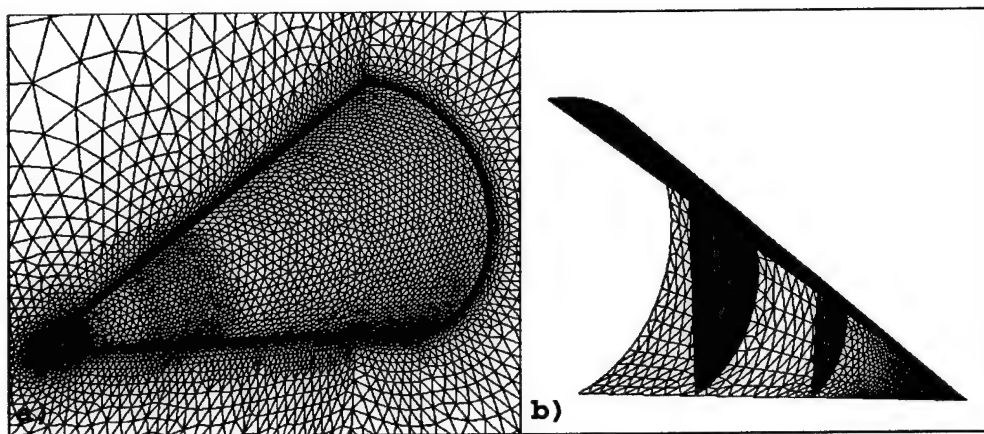

Figure 2a Nose-Cone: Surface Grids for CFD and CSD/CTD

7.3 <u>Fragmenting Weapon</u>: The third case considered is that of a fragmenting weapon. The detonation and shock propagation was modeled using a JWL equation of state with FEFLO98. The structural response, which included tearing and failure of elements, was computed using GA-DYNA, General Atomics' version of DYNA3D. At the beginning, the walls of the weapon separate two flow domains: the inner one, consisting of high explosive, and the outer one, consisting of air. As the structure of the weapon begins to fail, fragments are shrunk and the ensuing gaps are automatically remeshed,

13

leading to one contuinuous domain. The topology reconstruction from the discrete data passed to FEFLO from GA-DYNA is completely automatic, requiring no user intervention at any stage of the simulation. The mesh in the fluid domain was adapted using sources for geometric fidelity and the modified H2-seminorm error indicator proposed in . The sources required for geometric fidelity are constructed automatically during the topology reconstruction from the CSD surface faces. At the end of the run, the flow domain contains approximately 750 independently flying bodies and 16 million elements. Figure 3 shows the development of the detonation. The fragmentation of the weapon is clearly visible. Figure 3c shows the correlation with the observed experimental evidence.
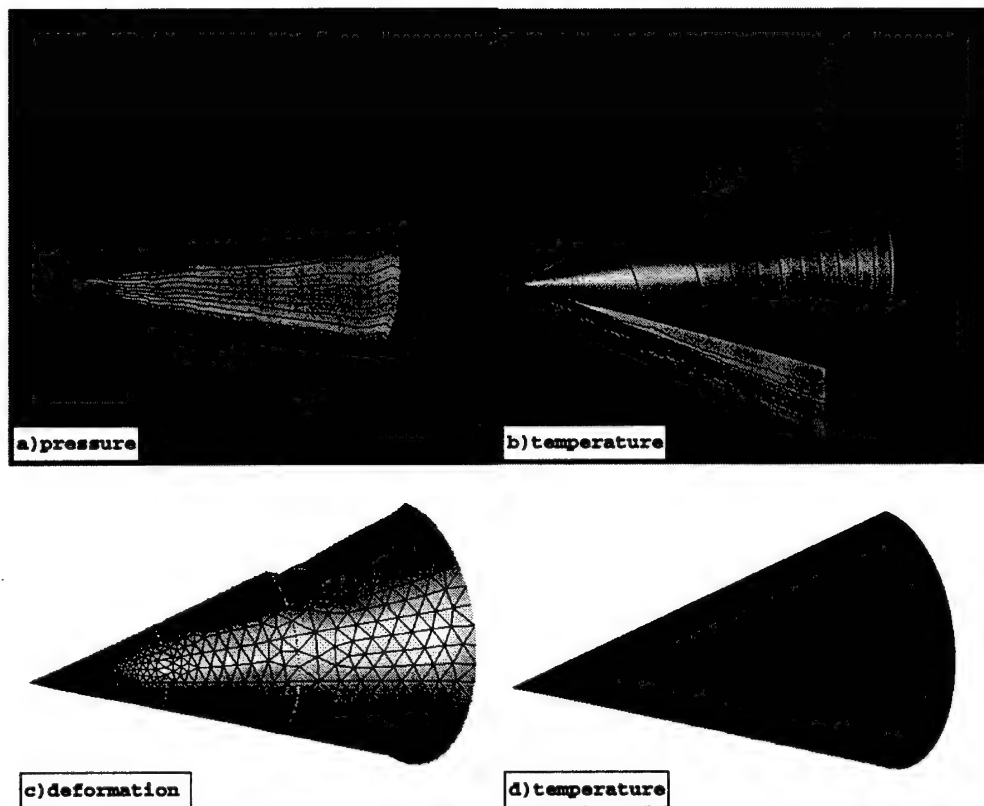

Figure 2b   Nose-Cone: CFD/CSD/CTD Results Obtained

7.4 <u>Blast Interaction With a Generic Ship Hull</u>: Figure 4 shows the interaction of an explosion with a generic ship hull. For this fully coupled CFD/CSD run, the structure was modeled with quadrilateral shell elements, the fluid as a mixture of high explosive and air, and mesh embedding was employed. The structural elements were assumed to fail once the average strain in an element exceeded 60%. As the shell elements failed, the fluid domain underwent topological changes. Figures 4a-d show the structure as well as the pressure contours in a cut plane at two times during the run. The influence

14

of bulkheads on surface velocity can clearly be discerned. Note also the failure of the structure, and the invasion of high pressure into the chamber. The distortion and inter-penetration of the structural elements is such that the traditional moving mesh approach (with topology reconstruction, remeshing, ALE formulation, remeshing, etc.) will invariably fail for this class of problems.
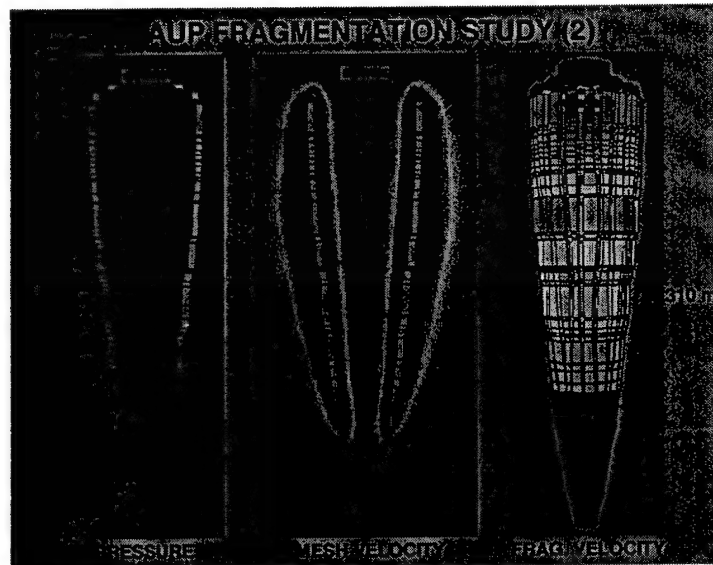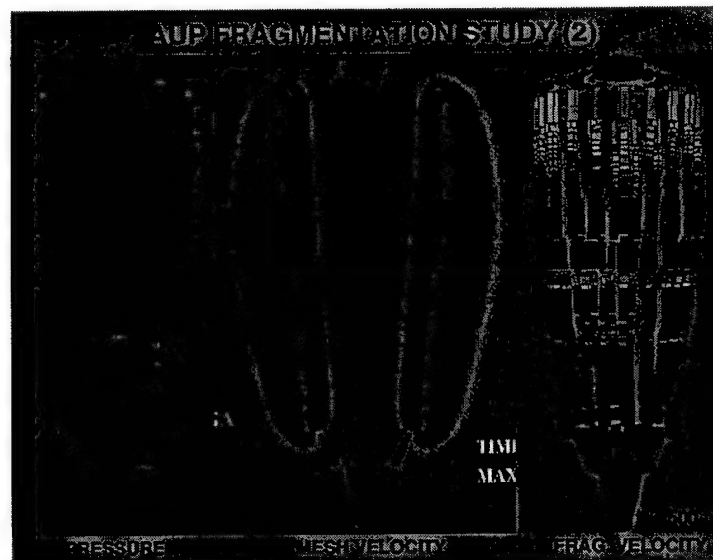

Figure 3a Fragmenting Weapon at 310msec

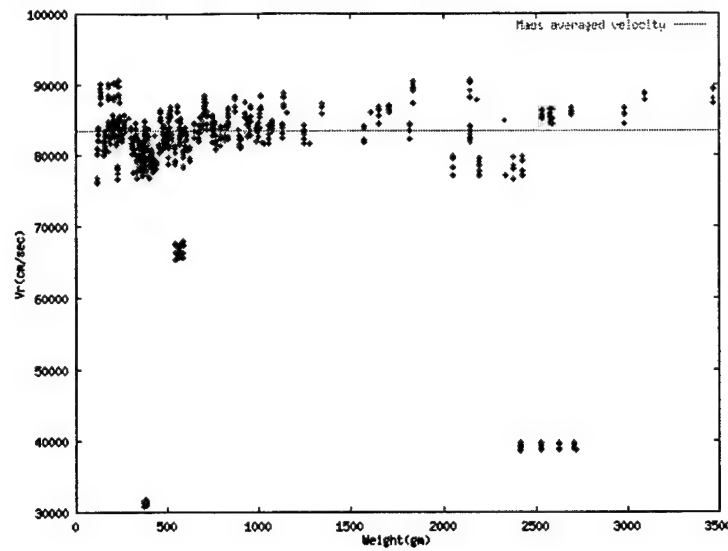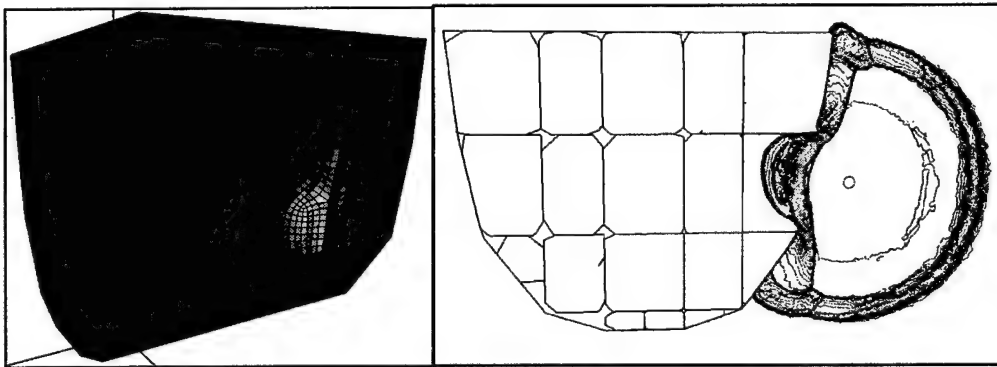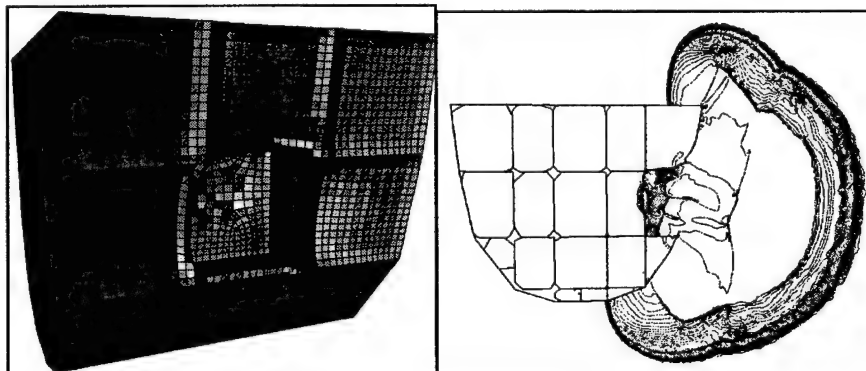Figure 3b Fragmenting Weapon at 500msec

15

Figure 3c Radial Velocity as a Function of Fragment Weight

Figures 4a,b: Surface and Pressure in Cut Plane at 20msec

Figures 4c,d: Surface and Pressure in Cut Plane at 50msec

16

8. CONCLUSIONS AND OUTLOOK

The methodologies and software required for Fluid-Structure-(Thermal) interaction simulations have progressed rapidly over the last decade. Several packages (e.g. FEFLO/GA-DYNA3D/COSMIC-NASTRAN) are offering the possibility of fully non-linear coupled CSD, CFD and CTD in a production environment. Looking towards the future, we envision a multidisciplinary, database-linked framework that is accessible from anywhere on demand, simulations with unprecedented detail and realism carried out in fast succession, virtual meeting spaces where geographically displaced designers and engineers discuss and analyze collaboratively new ideas, and first-principles driven virtual reality.

9. ACKNOWLEDGEMENTS

10. REFERENCES

- R. Löhner, Chi Yang, E. Oñate and S. Idelssohn - An Unstructured Grid-Based, Parallel Free Surface Solver; *Appl. Num. Math.* 31, 271-293 (1999).

- R. Löhner, C. Yang, J. Cebral, J.D. Baum, H. Luo, E. Mestreau, D. Pelessone and C. Charman - Fluid-Structure Interaction Algorithms for Rupture and Topology Change; *Proc. 1999 JSME Computational Mechanics Division Meeting*, Matsuyama, Japan, November (1999).

- R. Löhner, Chi Yang, J.D. Baum, H. Luo, D. Pelessone and C. Charman - The Numerical Simulation of Strongly Unsteady Flows With Hundreds of Moving Bodies; *Int. J. Num. Meth. Fluids* 31, 113-120 (1999).

- R. Ramamurti, W.C. Sandberg and R. Löhner - Computation of Unsteady Flow Past Deforming Geometries; *Int. J. Comp. Fluid Dyn.* , 83-99 (1999).

- R. Löhner - A Parallel Advancing Front Grid Generation Scheme; *AIAA*-00-1005 (2000).

- R. Löhner - Advances in Unstructured Grid Generation; *Proc. of ECCOMAS 2000 Conf.* , Barcelona, Spain, September (2000).

- R. Löhner, O. Soto and F. Camelli - On Implicit Projection Schemes; *4th Annual GMU Transport and Dispersion Modeling Workshop*, Fairfax, VA, July (2000).

- O. Soto and R. Löhner - CFD Optimization Using an Incomplete-Gradient Adjoint Approach; *AIAA*-00-0666 (2000).

- R. Ramamurti, W.C. Sandberg and R. Löhner - Simulation of the Dynamics of Micro Air Vehicles; *AIAA*-00-0896 (2000). *AIAA*-00-0666 (2000).

17

- J.R. Cebral, R. Löhner and J. Burgess - Computer Simulation of Cerebral Artery Clipping: Relevance to Aneurysm Neuro-Surgery Planning; *Proc. of ECCOMAS 2000 Conf.* , Barcelona, Spain, September (2000).

- C. Yang, F. Noblesse R. Löhner and D. Hendrix - Practical CFD Applications to Design of a Wave Cancellation Multihull Ship; *Proc. of the 23rd Symp. on Naval Hydrodynamics*, Val de Reuil, France, September (2000).

- C. Yang, R. Löhner, F. Noblesse and T.T. Huang - Calculation of Ship Sinkage and Trim Using Unstructured Grids; *Proc. of ECCOMAS 2000 Conf.* , Barcelona, Spain, September (2000).

- C. Yang, R. Löhner and F. Noblesse - Farfield Extension of Nearfield Steady Ship Waves; *Ship Technology Research* 47, 1, 22-34 (2000).

- J. Waltz and R. Löhner - A Grid Coarsening Algorithm for Unstructured Multigrid Applications; *AIAA*-00-0925 (2000).

- H. Luo, D. Sharov, J.D. Baum and R. Löhner - On the Computation of Compressible Turbulent Flows on Unstructured Grids; *AIAA*-00-0926 (2000).

- D. Sharov, H. Luo, J.D. Baum and R. Löhner - Implementation of Unstructured Grid GMRES+LU-SGS Method on Shared-Memory, Cache-Based Parallel Computers; *AIAA*-00-0927 (2000).

- R. Löhner - A Parallel Advancing Front Grid Generation Scheme; *AIAA*-00-1005 (2000).

- R. Löhner and J. Cebral - Generation of Non-Isotropic Unstructured Grids via Directional Refinement; *Int. J. Num. Meth. Eng.* 49, 219-232 (2000).

- R. Löhner, Chi Yang, J. Cebral, O. Soto, F. Camelli, J.D. Baum, H. Luo, E. Mestreau, D. Sharov, R. Ramamurti, W. Sandberg and Ch. Oh - Advances in FEFLO; *AIAA*-01-0592 (2001).

- H. Luo, J.D. Baum and R. Löhner - An Accurate, Fast, Matrix-Free Implicit Method for Computing Unsteady Flows on Unstructured Grids; *Comp. and Fluids* 30, 137-159 (2001).

- R. Ramamurti, W.C. Sandberg and R. Löhner - Simulation of Flow About Flapping Airfoils Using a Finite Element Incompressible Flow Solver; *AIAA J.* 39, 2, 253-260 (2001).

- R. Löhner - A Parallel Advancing Front Grid Generation Scheme; *Int. J. Num. Meth. Eng.* 51, 663-678 (2001).

- O. Soto and R. Löhner - CFD Shape Optimization Using an Incomplete-Gradient Adjoint Formulation; *Int. J. Num. Meth. Eng.* 51, 735-753 (2001).

- J.R. Cebral and R. Löhner - From Medical Images to Anatomically Accurate Finite Element Grids; *Int. J. Num. Meth. Eng.* 51, 985-1008 (2001).

- J.R. Cebral, R. Löhner, P.L. Choyke and P.J. Yim - Merging of Intersecting Triangulations for Finite Element Modeling; *J. of Biomechanics* 34, 815-819 (2001).

- H. Luo, J.D. Baum and R. Löhner - A Fast, Matrix-Free Implicit Method for Computing Low Mach Number Flows on Unstructured Grids; *Int. J. CFD* 14, 133-157 (2001).

- H. Luo, D. Sharov, J.D. Baum and R. Löhner - On the Computation of Compressible Turbulent Flows on Unstructured Grids; *Int. J. CFD* 14, 253-270 (2001).

- J. Cebral and R. Löhner - Flow Visualization on Unstructured Grids Using Geometrical Cuts, Vortex Detection and Shock Surfaces; *AIAA*-01-0915 (2001).

- R. Löhner - Overlapping Unstructured Grids; *AIAA*-01-0439 (2001).

- O. Soto and R. Löhner - General Methodologies for Incompressible Flow Design Problems; *AIAA*-01-1061 (2001).

- D. Sharov, H. Luo, J.D. Baum and R. Löhner - Unstructured Navier-Stokes Grid Generation at Corners and Ridges; *AIAA*-01-2600 (2001).

- H. Luo, D. Sharov, J.D. Baum and R. Löhner - An Overlapping Unstructured Grid Method for Viscous Flows; *AIAA*-01-2604 (2001).

- O. Soto, R. Löhner and J.R. Cebral - An Implicit Monolithic Time Accurate Finite Element Scheme for Incompressible Flow Problems; *AIAA*-01-2616-CP (2001).

- J.D. Baum, H.Luo, E.L. Mestreau, D. Sharov, R. Löhner, D. Pelessone and Ch. Charman - Recent Developments of a Coupled CFD/CSD Methodology; *AIAA*-01-2618 (2001).

- J.D. Baum, H.Luo, E.L. Mestreau, D. Sharov, R. Löhner, D. Pelessone and Ch. Charman - A New Coupled CFD/CSD Methodology; Paper presented at ICFD, March (2001).

- C. Yang, F. Noblesse and R. Löhner - Verification of Fourier-Kochin Representation of Waves; *Proc. 16th Int. Workshop on Water Waves and Floating Bodies*, Hiroshima, Japan, April (2001).

- J.D. Baum, H.Luo, E.L. Mestreau, D. Sharov, R. Löhner, D. Pelessone and Ch. Charman - Recent Developments of a Coupled CFD/CSD Methodology for Simulating Structural Response to Airblast and Fragment Loading; Paper presented at ICCS 2001, San Francisco, CA, May (2001).

- J.R. Cebral, R. Löhner, O. Soto and P.J. Yim - On the Modeling of Carotid Artery Blood Flow From Magnetic Resonance Images; *Proc. ASME-BED Bioengineering Conf.* , Vol. 50, June 27-July 1 (2001).

19

- J.D. Baum, H.Luo, E.L. Mestreau, D. Sharov, R. Löhner, D. Pelessone and Ch. Charman - Some New Developments of a Coupled CFD/CSD Methodology; Shanghai, China, July (2001).

- R. Löhner and Chi Yang - Tracking Vortices Over Large Distances Using Vorticity Confinement; paper presented at *ECCOMAS CFD 2001 Conf.*, Swansea, Wales, September (2001).

- J.R. Cebral, R. Löhner and P. Yim - Image-Based Blood Flow Modeling as a Predictive Tool for Planning Medical Interventions; paper presented at *ECCOMAS CFD 2001 Conf.*, Swansea, Wales, September (2001).

- O. Soto, R. Löhner, J.R. Cebral and R. Codina - A Time-Accurate Implicit-Monolithic Finite Element Scheme for Incompressible Flow Problems; paper presented at *ECCOMAS CFD 2001 Conf.*, Swansea, Wales, September (2001).

- D. Sharov, J.D. Baum, H. Luo and R. Löhner - Recent Advances in RANS Gridding; paper presented at *ECCOMAS CFD 2001 Conf.*, Swansea, Wales, September (2001).

- C. Yang, F. Noblesse and R. Löhner - Practical Hydrodynamic Optimization of a Trimaran; *SNAME Annual Meeting*, Orlando, FL., October (2001).

- P.J. Yim, J.R. Cebral, V.B. Ho, H.B. Marcos, G. Boudewijn, C. Vasbinder, A. Rowedder, A. Sortur, J.M.A. van Engelshoven, O. Soto, R. Löhner and P.L. Choyke - Computational Modeling of Blood Flow Patterns from Magnetic Resonance Imaging, *Proc. RSNA*, October (2001).

- J.R. Cebral, R. Löhner, O. Soto, P.L. Choyke and P.J. Yim - Patient-Specific Simulation of Carotid Artery Stenting Using Computational Fluid Dynamics, pp. 153-160 in *Springer LNCS 2208* (Proc. MICCAI 2001, Utrecht, The Netherlands, Oct. 14-17) (2001).